

Engineering Software Solutions
Technical Software Company

Engineering Software Solutions



2D Frame Analysis

Static and dynamic analysis of frames and trusses

v2.0

User manual

GENERAL

Thank you for using Engissol products. Our R&D department which constitutes of experienced engineers and programmers can assure you that you will be provided with a finite element application of high standards and guaranteed accurate analysis results. Engissol's know-how with respect to engineering application development is incomparable, a fact that is proven by thousands of satisfied end-users around the world.

2D Frame Analysis is a finite element application that can perform static and dynamic analysis of plane frames and trusses with any type of complexity. It uses the powerful and well-known analysis core "ENGISSOL – FEM Library" which has received many awards because of its flexibility in handling large-scale linear and non-linear arithmetic problems. 2D Frame Analysis is intentionally limited to linear static and dynamic analysis in order to cover the everyday engineering demand efficiently. It has been tested thoroughly to meet specific standards and is considered to be the best frame and truss analysis application at a very low price.

Apart from this, at Engissol we are always looking for feedback on how to improve our products. You can contact us at engissol@engisso.com.

PROGRAM FEATURES

The Frame Analysis application is available in 3 different packages (Static, Dynamic and Truss).

Static Analysis version: Static analysis of all kind of frames

Truss analysis version: Static analysis of all kind of trusses


















Dynamic analysis version: Static and dynamic analysis of all kind of frames and trusses

Each licensed feature from the following is activated. **Features in red are only available in the Dynamic Analysis version.**








A. Static Edition

Features

Analysis

-  Use of highly flexible, general, finite element method
-  Static and dynamic analysis of multi span beams, 2D trusses and 2D frames
-  **Modal analysis and eigenvalues estimation**
-  **User controlled damping coefficient**
-  **Mode superposition method in order to calculate the dynamic response**
-  **Automatic creation of consistent mass matrix (Engissol's patent)**
-  **Multi-step dynamic analysis which is fully customized**
-  **User controlled step and time parameters**
-  **Definition of time-dependent loading on elements and nodes**
-  Unlimited number of Nodes and Beams
-  3 Degrees of freedom per Node, 6 per Beam
-  Supports all major measure units
-  All type of boundary conditions (fixed, rollers, etc.)
-  Translational and rotational spring supports
-  Initial displacement and speed conditions
-  Extremely easy model creation, no need to first define nodes and then elements, nodes are produced automatically
-  Consideration of thermal loads

Pre-processing:

-  Top quality graphics rendering
-  Full GUI including zoom, pan, grid, snap options
-  Every user action can be done graphically at real time
-  Immediate switch to other measure units
-  Easy model creation (members, loads, supports etc) in a user friendly environment with many graphical features
-  Embedded library with major steel section shapes
-  Definition of custom beam sections

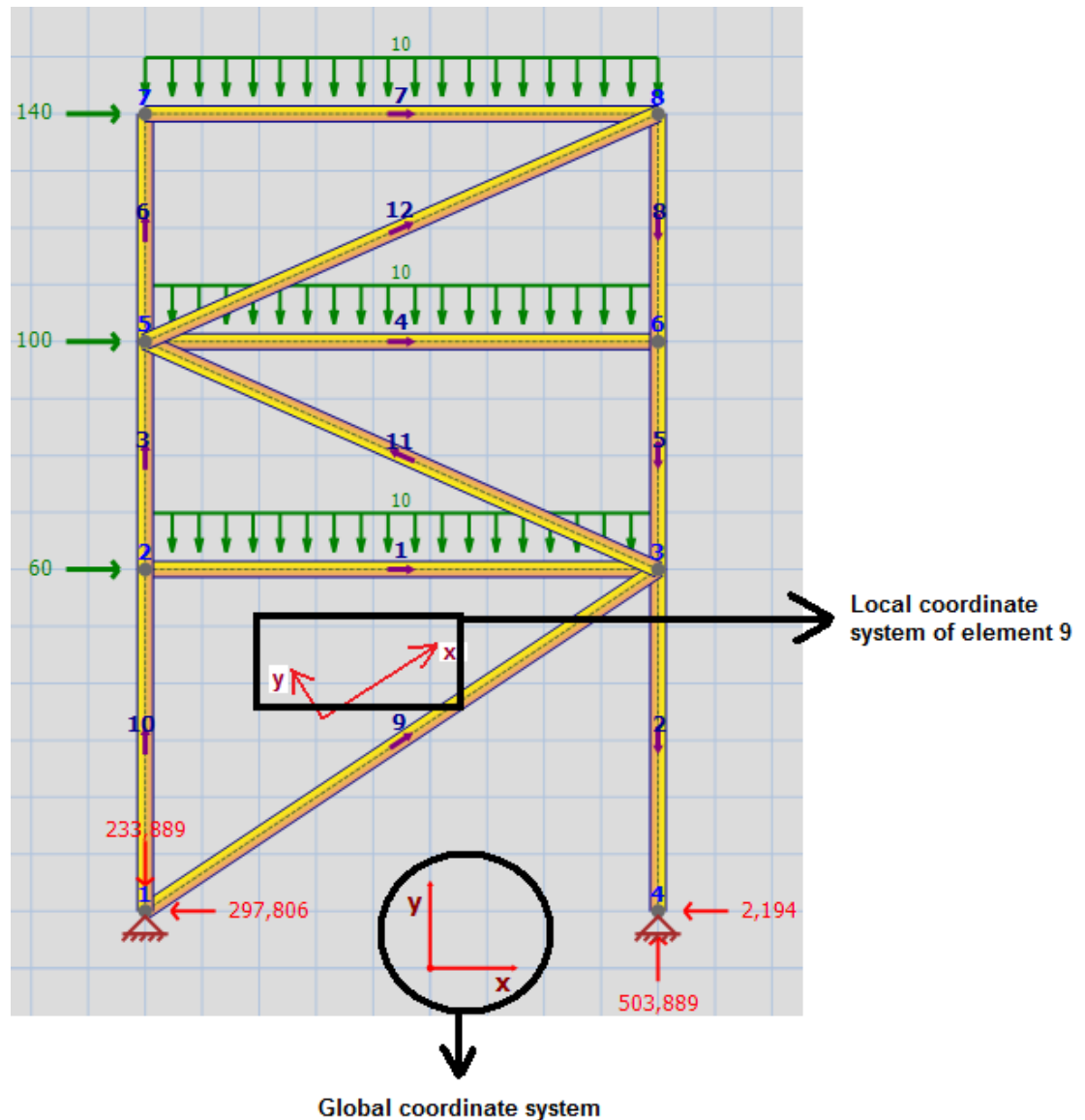
- ✚ Embedded library with major materials (wood, steel, concrete)
- ✚ Definition of custom materials
- ✚ Easy and fast modification of the material properties
- ✚ End release option at specified degrees of freedom (all types of end releases are supported)
- ✚ Quick apply vertical and horizontal distributed loads on elements
- ✚ **Application of time dependent loads at each time step graphically**


Post-processing:

- ✚ Calculation and drawing of the deformed shape of the given model
- ✚ Analytical view of the results of analysis, the geometry, the static model, the loads etc.
- ✚ Drawing of axial - shear force diagrams and bending moment diagram
- ✚ Scaling to above diagrams and the deformed model interactively
- ✚ Internal forces calculation at each location along the members
- ✚ Stress calculation at the top and bottom fiber along each member
- ✚ **Graphic representation of every model parameter (internal forces, moments, displacements, rotations, reactions) vs. time**
- ✚ **Drawing axial - shear force and bending moment diagrams at each time step**
- ✚ **Drawing of deformed shape at each time step**
- ✚ High quality analysis reports with all analysis results, including stress, internal forces and displacements diagrams, as well as drawings of the static model, its deformed shape and the axial-shear forces and moment diagrams
- ✚ Analysis report can be exported to pdf, rtf, xls and xml formats or it can be directly be printed
- ✚ All model data are exported to access database

COORDINATE SYSTEMS

Two coordinate systems are used by the program. The local one and the global one. The global system is the same for the whole model. The local system is defined by placing its x axis on the examined element pointing from the smaller to the greater edge node number. This convention is respected at every reference to local and global coordinate systems.

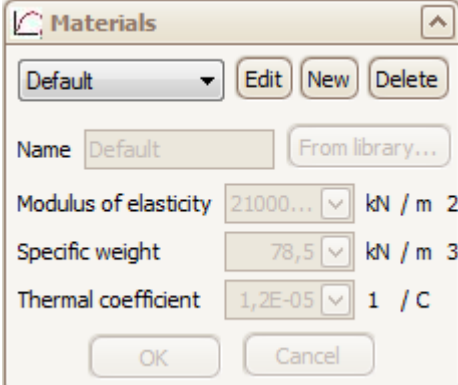


Please note that the positive rotation convention is the counter clockwise  for both coordinate systems.

MENU DESCRIPTION

Material tab

You can define materials in this tab. Each material is defined by the following properties:

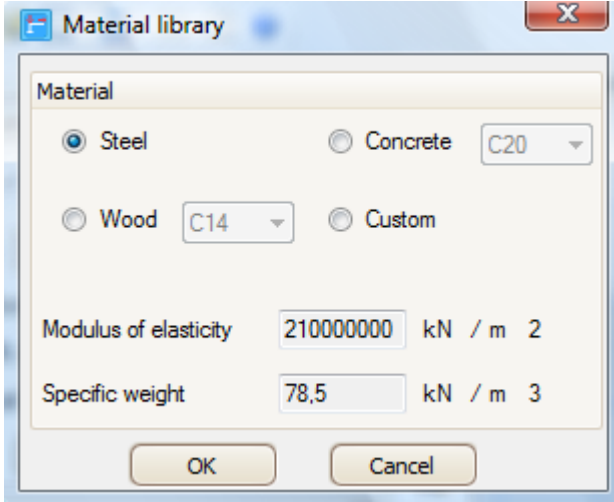


The 'Materials' dialog box is used to define material properties. It features a 'Default' dropdown menu, 'Edit', 'New', and 'Delete' buttons. Below these are input fields for 'Name' (set to 'Default'), 'Modulus of elasticity' (21000...), 'Specific weight' (78,5), and 'Thermal coefficient' (1,2E-05). A 'From library...' button is also present. At the bottom are 'OK' and 'Cancel' buttons.

- ✚ a name
- ✚ the modulus of elasticity
- ✚ the specific weight (this value is used only during dynamic analysis in order to calculate the mass matrix)
- ✚ the thermal coefficient, which corresponds to the axial strain when a unary temperature change is applied to a specimen of this material

Remember that the Default material cannot be deleted.

By clicking the “From library” button, the following form is displayed which gives you the ability to insert a pre-defined material. Steel, concrete (all grades), wood (all grades) and custom material definition are supported.



The 'Material library' dialog box allows selecting a pre-defined material. It has radio buttons for 'Steel', 'Concrete', 'Wood', and 'Custom'. 'Steel' is selected. There are dropdown menus for 'Concrete' (C20) and 'Wood' (C14). Below are input fields for 'Modulus of elasticity' (210000000) and 'Specific weight' (78,5). 'OK' and 'Cancel' buttons are at the bottom.

Section tab

Sections are defined in this tab. The following properties are needed in order to define a section:

- ✚ a name
- ✚ the cross section area
- ✚ the moment of inertia (this inertia moment corresponds to an axis pointing into the screen)
- ✚ the section height, which is used during the calculation of the bending moments on the frame element due to temperature differences above and below it

Remember that the Default section cannot be deleted.

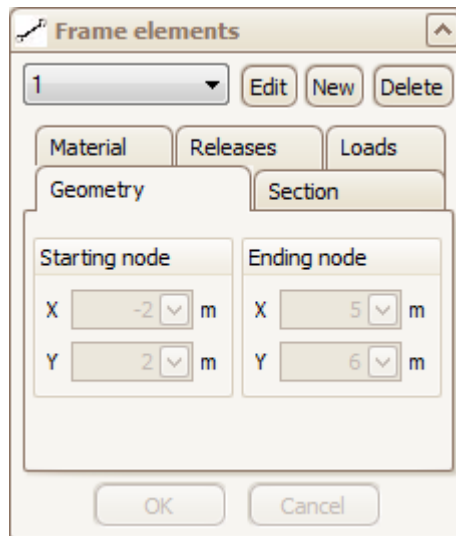
By clicking the “From library” button, the following form is displayed which gives you the ability to insert standard cross sections.

Frame element tab


In this tab, all needed data to define a frame element are given.

Geometry page

The geometry of both, starting and ending nodes, is given in this page.

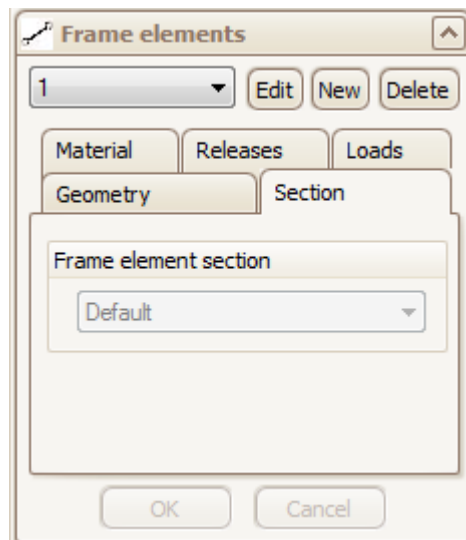


The dialog box is titled "Frame elements". It has a dropdown menu showing "1" and three buttons: "Edit", "New", and "Delete". Below these are three tabs: "Material", "Releases", and "Loads". The "Geometry" tab is selected, and it contains two sub-tabs: "Geometry" and "Section". The "Geometry" sub-tab is active, showing fields for "Starting node" and "Ending node". Under "Starting node", there are two rows: "X" with a value of "-2" and "Y" with a value of "2", both followed by a unit "m". Under "Ending node", there are two rows: "X" with a value of "5" and "Y" with a value of "6", both followed by a unit "m". At the bottom are "OK" and "Cancel" buttons.

You can define a frame element by clicking on the “New” button, or you can insert it graphically by using the  button of the program toolbar.

Section page

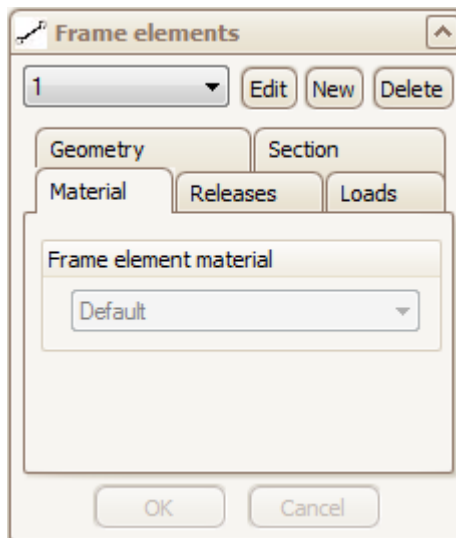
You can choose here the section of the frame element that you are about to define.



The dialog box is titled "Frame elements". It has a dropdown menu showing "1" and three buttons: "Edit", "New", and "Delete". Below these are three tabs: "Material", "Releases", and "Loads". The "Section" tab is selected, and it contains a sub-tab: "Section". The "Section" sub-tab is active, showing a dropdown menu labeled "Frame element section" with the value "Default". At the bottom are "OK" and "Cancel" buttons.

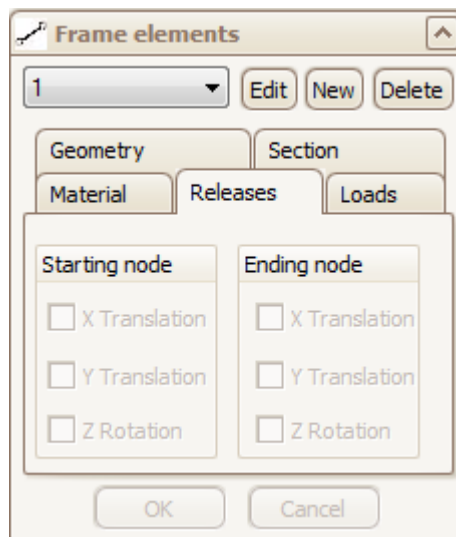
Material page

You can choose here the material of the frame element that you are about to define.



Release page

The end releases of the frame element are defined in this page. They all refer to the local coordinate system of the frame element.








Loads page

Distributed loads on the frame element are defined in this page. They refer to the local coordinate system of the frame element.

Node tab

As described, the nodes are generated automatically after the creation of each frame element. In this tab, the properties of the generated nodes are given as following:

-  **Coordinates:** The coordinates of the node in the global coordinate system.
-  **Nodal loads:** The loads applied to the node in the global coordinate system.
-  **Supports:** The supports of the node.
 In case of springs, the full support option should be unchecked for the corresponding degree of freedom. The related spring constant box will then be enabled. You have to give then the constant of the matching spring (translational or rotational). Remember all these options refer to the global coordinate system.
-  **Initial displacements:** The initial, if any, displacements (for degrees of freedom x,y) and rotation (for degree of freedom z) are given in this section. They are taken into account during the static and the dynamic analysis.
-  **Initial velocities for dynamic analysis:** The initial, if any, velocities of the node are given in this section. They are used to define the initial conditions of the model during the dynamic analysis and they refer to the global coordinate system.

Auto generated nodes

1

Edit

Coordinates

X

-2

m
Y

2

m

Nodal Loads

Px

0

kN
Py

0

kN
Mz

0

kN m

Supports

Degree of freedom	Full support	Spring constant
<input type="checkbox"/> X Translation	<input checked="" type="checkbox"/>	<div>0</div> kN / m
<input type="checkbox"/> Y Translation	<input checked="" type="checkbox"/>	<div>0</div> kN / m
<input type="checkbox"/> Z Rotation	<input checked="" type="checkbox"/>	<div>0</div> kN m /rad

Initial displacements

Degree of freedom	Initial displacements/rotation
<input type="checkbox"/> X Translation	<div>0</div> m
<input type="checkbox"/> Y Translation	<div>0</div> m
<input type="checkbox"/> Z Rotation	<div>0</div> rad

Initial velocities for dynamic analysis

Degree of freedom	Initial velocities
<input type="checkbox"/> X Translation	<div>0</div> m /sec
<input type="checkbox"/> Y Translation	<div>0</div> m /sec
<input type="checkbox"/> Z Rotation	<div>0</div> rad /sec

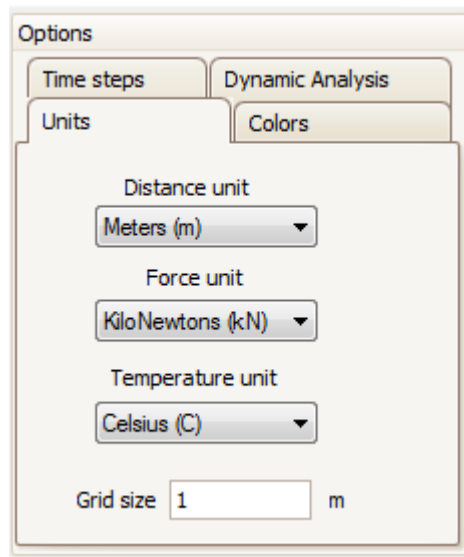
OK

Cancel

Options tab

In this tab, general program options are defined.

Units page



The 'Options' dialog box has four tabs: 'Time steps', 'Dynamic Analysis', 'Units', and 'Colors'. The 'Units' tab is selected. It contains three dropdown menus for 'Distance unit' (set to 'Meters (m)'), 'Force unit' (set to 'KiloNewtons (kN)'), and 'Temperature unit' (set to 'Celsius (C)'). At the bottom, there is a 'Grid size' input field with the value '1' and a unit 'm'.

The measure units are defined here. You can select among a wide variety of units. The grid size is also defined here.

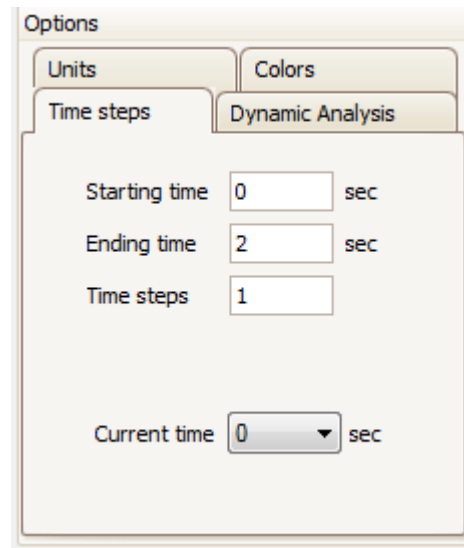
Colors page



The 'Options' dialog box has four tabs: 'Time steps', 'Dynamic Analysis', 'Units', and 'Colors'. The 'Colors' tab is selected. It contains two columns of buttons for customizing drawing colors. The left column includes: 'Background', 'Grid', 'Axes', 'Nodes labels', 'Nodal loads', 'Elem. labels', and 'Elem. Axis'. The right column includes: 'Elem. loads', 'Elem. fill 1', 'Elem. fill 2', 'Springs', 'Supports', 'Reactions', and 'Deformed'.

From this page, drawing colors can be customized.

Time steps page

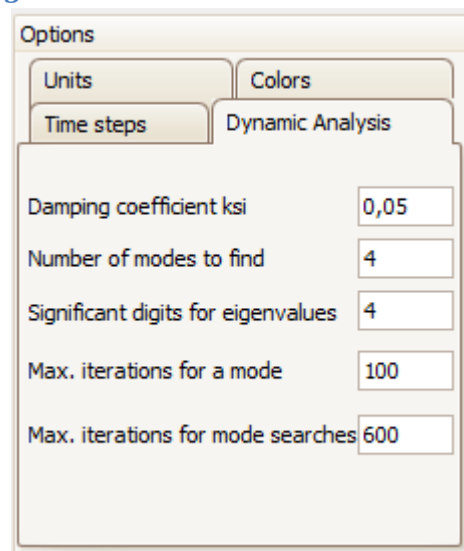


The screenshot shows the 'Options' dialog box with the 'Time steps' tab selected. The 'Dynamic Analysis' sub-tab is also active. The following parameters are visible:

Parameter	Value	Unit
Starting time	0	sec
Ending time	2	sec
Time steps	1	
Current time	0	sec

You can define here parameters that are used during the dynamic analysis, such as the time period and the time step number. These options affect only the dynamic analysis procedure. By setting the current time parameter, you can see your model (both loads and results) at the selected time.





Dynamic preferences page



The screenshot shows the 'Options' dialog box with the 'Dynamic Analysis' sub-tab selected. The following parameters are visible:

Parameter	Value
Damping coefficient ksi	0,05
Number of modes to find	4
Significant digits for eigenvalues	4
Max. iterations for a mode	100
Max. iterations for mode searches	600

The following dynamic preferences are defined in this page. They only affect the dynamic analysis procedure:

-  **Damping coefficient ksi:** The damping coefficient that is applied to each mode as modal damping. (equal for all modes) Zero value means no damping. Ksi mainly depends on the material of the structure.
-  **Number of modes to find:** The desired number of nodes to be found.
-  **Significant digits for eigenvalues:** Decimal accuracy of the calculated eigenvalues.
-  **Max. iterations for a mode:** Maximum number of iterations for the estimation of a mode.

🚦 **Max. iterations for mode searches:** Maximum searching iteration number to estimate a mode.

It is suggested not to change the last three parameters in order to avoid arithmetic and convergence problems during the dynamic analysis.

Results tab

In this tab, all analysis results regarding nodes and frame elements are presented. **In case of dynamic analysis, these results refer to the times step selected at the Time steps page in the Options tab.**

Nodes page

Nodal analysis results are presented in this section. For the selected node, the displacements (x, y) and rotation (z) are shown, as well as the reactions, if any. In case of spring supports, the corresponding spring reactions are shown. All these results refer to the global coordinate system of the model.

The screenshot shows a software window titled "Results" with two tabs: "Nodes" and "Frame elements". The "Nodes" tab is active. Below the tabs is a "Node" dropdown menu set to "1". Below this, there are nine rows of results, each with a label, a text input field, and a unit. The values are as follows:

Label	Value	Unit
X Translation	0	m
Y Translation	0	m
Z Rotation	0	rad
Reaction X	-5,68434...	kN
Reaction Y	80,0000...	kN
Reaction Z	319,999...	kN m
Spring reaction X	0	kN
Spring reaction Y	0	kN
Spring reaction Z	0	kN m

Frame elements page

Frame elements analysis results are presented in this section. For the selected frame element, the edge node displacements/rotation are shown, as well as the internal forces acting on the edge nodes of the element. While all displacements and rotations refer to the local coordinate system of the element, the internal forces are presented respecting the classic statics convention. By clicking on the Internal forces and stresses button, you can get the internal forces and stresses at locations along the element length.

Results

Nodes Frame elements

Frame element

Results on Starting Node

Axial force	<input type="text" value="-80,0000..."/>	kN
Shear force	<input type="text" value="5,68434..."/>	kN
Moment	<input type="text" value="-319,999..."/>	kN m
X Translation	<input type="text" value="0"/>	m
Y Translation	<input type="text" value="0"/>	m
Z Rotation	<input type="text" value="0"/>	rad

Results on Ending Node

Axial force	<input type="text" value="-80,0000..."/>	kN
Shear force	<input type="text" value="5,68434..."/>	kN
Moment	<input type="text" value="-319,999..."/>	kN m
X Translation	<input type="text" value="-0,00022..."/>	m
Y Translation	<input type="text" value="-0,91428..."/>	m
Z Rotation	<input type="text" value="-0,30476..."/>	rad

[Internal forces and stresses...](#)

Internal forces and stresses along member

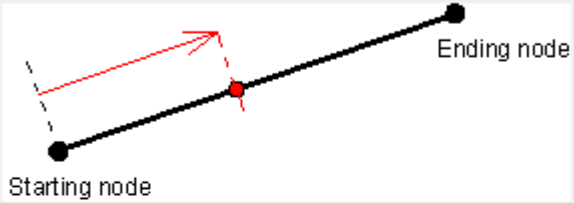
By choosing the desired frame element and the distance from the starting node, you can get internal element forces (axial-shear forces and bending moment) at this location. Normal and shear stresses are also calculated at this location.

Please note that internal forces are displayed in the local coordinate system of the element.

Internal forces and stresses along member

Member's id: 2

Distance from starting node: 4 m



Starting node

Ending node

Internal forces

Axial Force	0	kN
Shear Force	40	kN
Bending Moment	-80	kN m

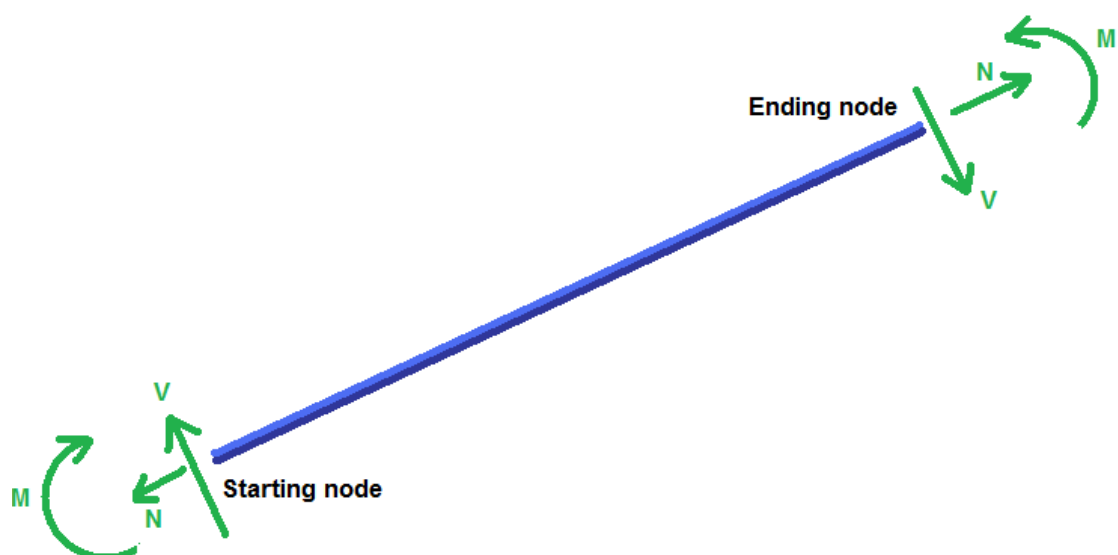
Stresses

Normal stress 1	-66666,667	kN / m ²
Normal stress 2	66666,667	kN / m ²
Shear stress	4000	kN / m ²

Close

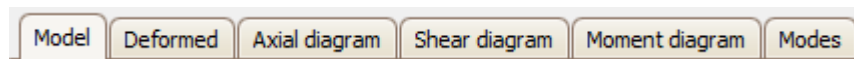
Classic statics convention

The internal forces are presented with respect to the following coordinate system which constitutes the classic statics convention:

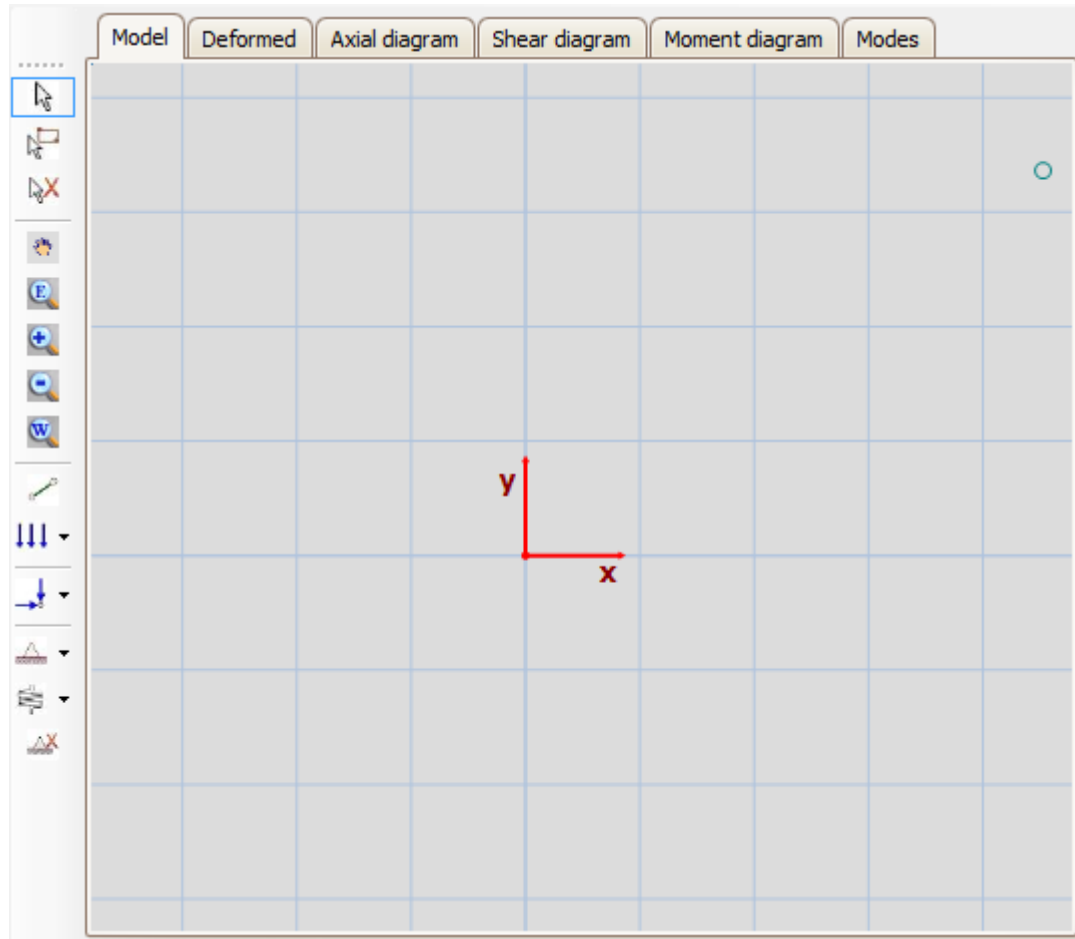


Main window description















The main window of the application includes the following tab page collection. A comprehensive description of each tab page follows.



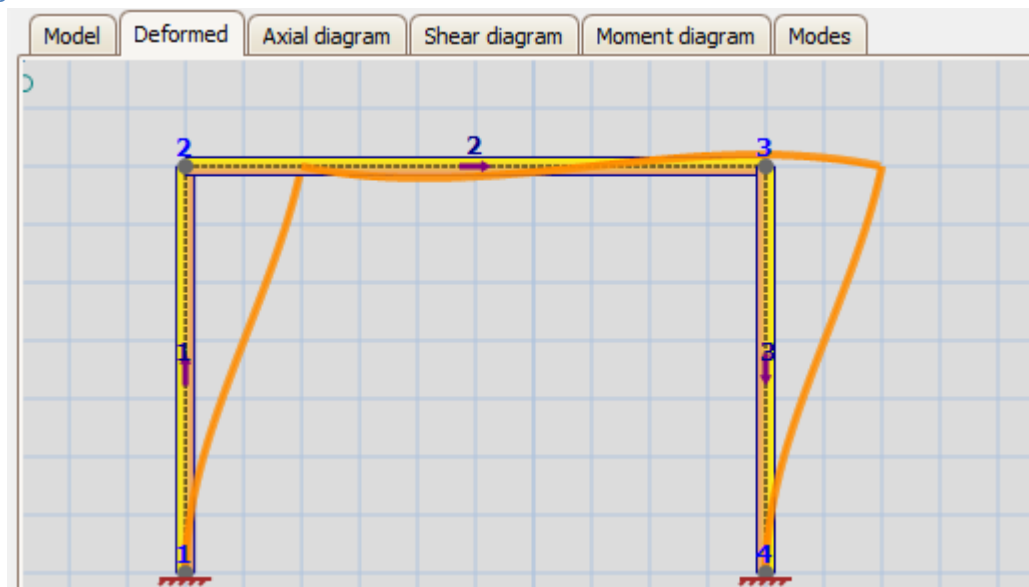
Model tab



The static model can very easily be created from this window. The toolbar on the left can help you during the drawing of the model interactively. A description of each toolbar button follows:

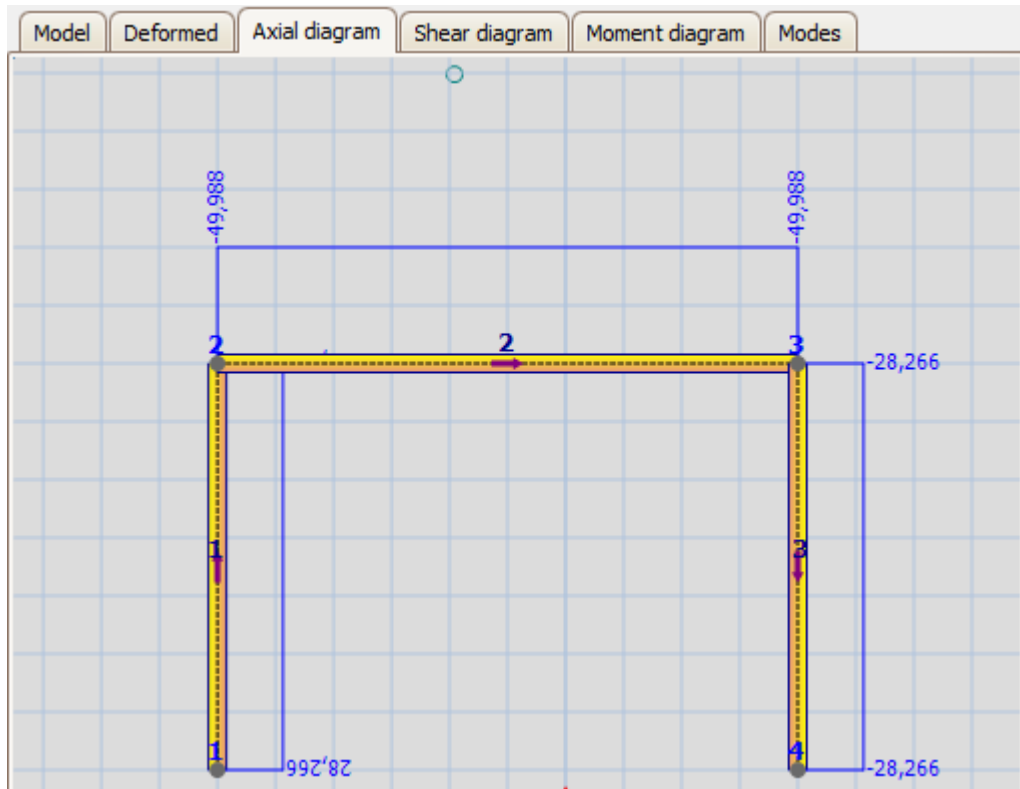
	Single selection
	Boxed selection
	Clear selection
	Pan
	Zoom extends
	Zoom in
	Zoom out
	Zoom window
	Draw an element
	Apply element load
	Apply nodal load
	Apply restraints (supports)
	Apply spring restraints (supports)
	Clear restraints (supports)

Deformed tab



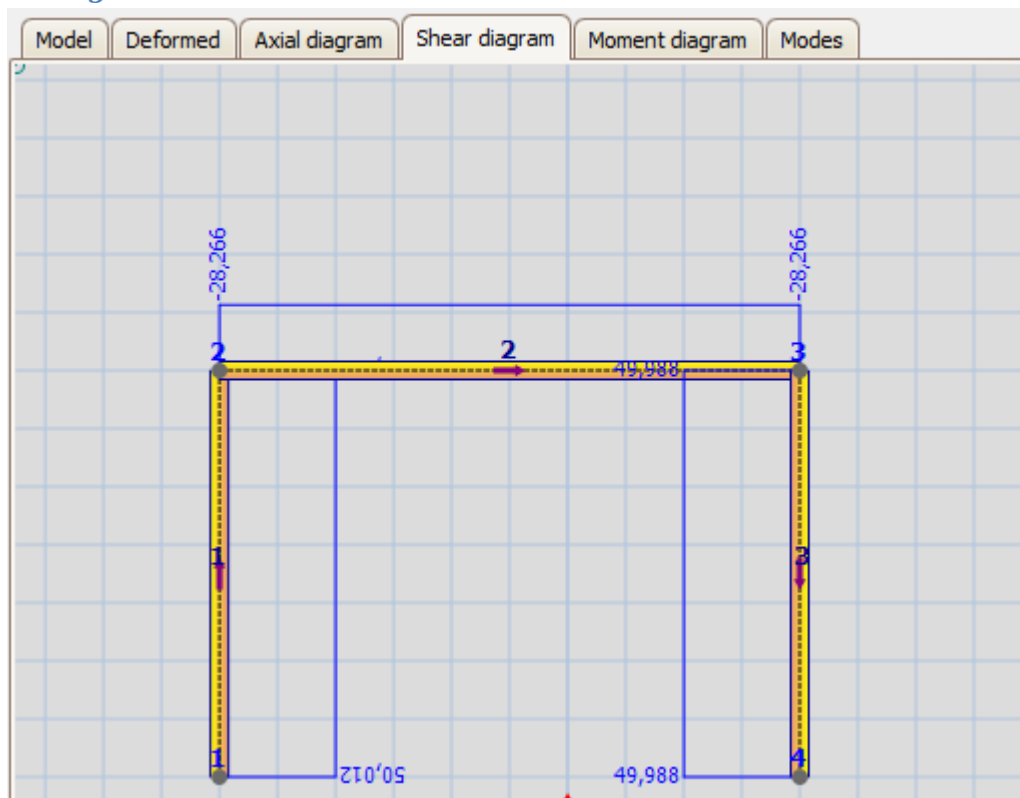
The deformed shape of the model is displayed in this tab.

Axial diagram tab



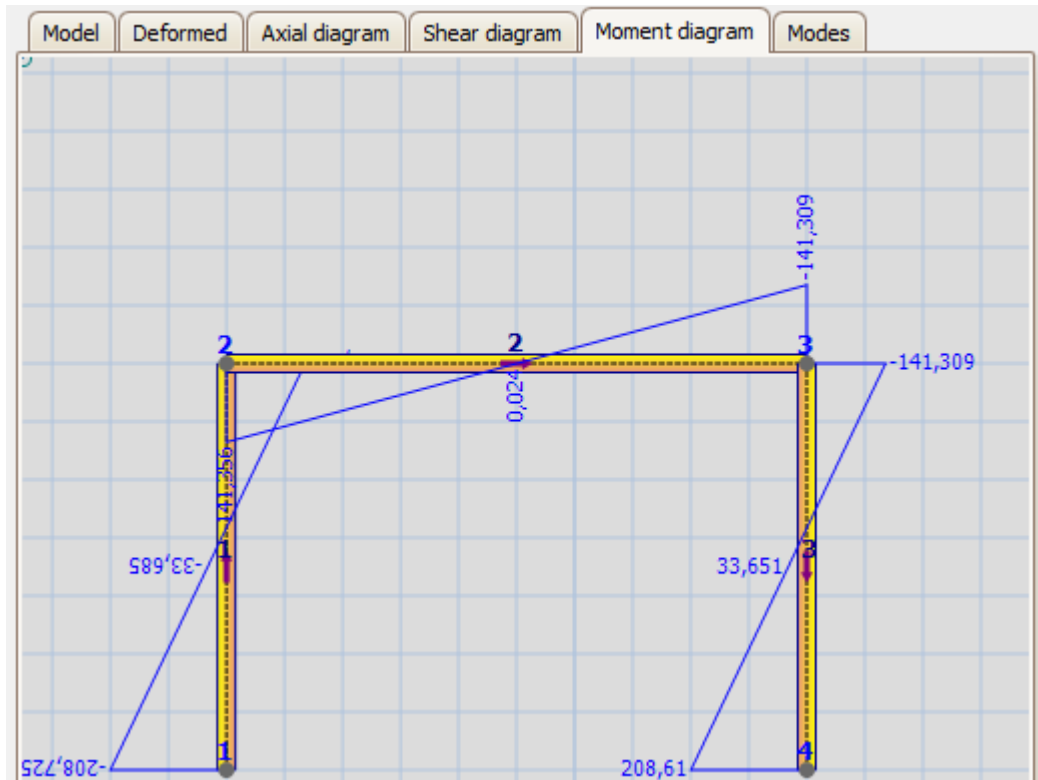
The axial force diagram is displayed in this tab.

Shear diagram tab



The shear force diagram is displayed in this tab.

Moment diagram tab



The moment diagram is displayed in this tab.

Modes tab



The calculated mode shapes are displayed in this tab. By clicking the Next button, more mode shapes are displayed. The calculated period for each mode is also displayed.


Status bar description

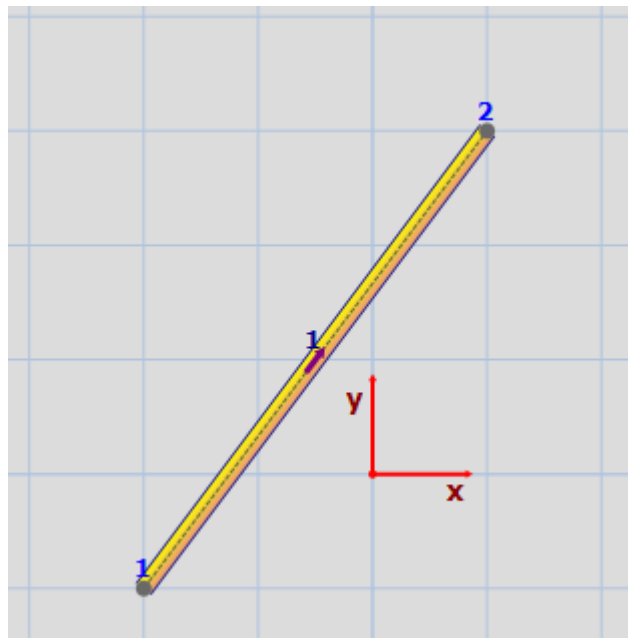
The status bar includes the following controls:

Draw grid	Snap to grid	Snap to nodes	Draw element loads	Draw nodal loads	Draw supports	Draw reactions	Show selected only	x= 6 m / y= 3 m
-----------	--------------	---------------	--------------------	------------------	---------------	----------------	--------------------	-----------------

Draw grid	If selected, the grid is displayed
Snap to grid	If selected, the mouse snaps to the grid points
Snap to nodes	If selected, the mouse snaps to all previously created nodes
Draw element loads	If selected, element loads are displayed
Draw nodal loads	If selected, nodal loads are displayed
Draw supports	If selected, supports are displayed
Draw reactions	If selected, reactions are displayed
Show selection only	If selected, only the selected elements are displayed


Drawing a frame element

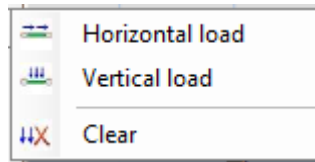
1. Click on the  icon
2. Move the mouse over the grid and click on the desired starting point.
3. Click on the corresponding ending point.



Please note that this could also be done from the Frame elements tab on the left program menu, under the Geometry page.

Applying loads on a frame element

1. Click on the  button and select from the following menu.

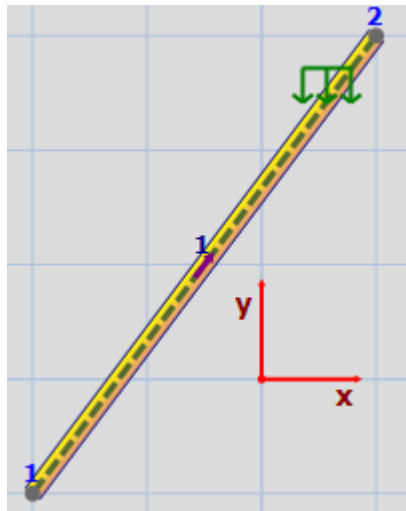


2. Enter a load value in the following form that will come up.

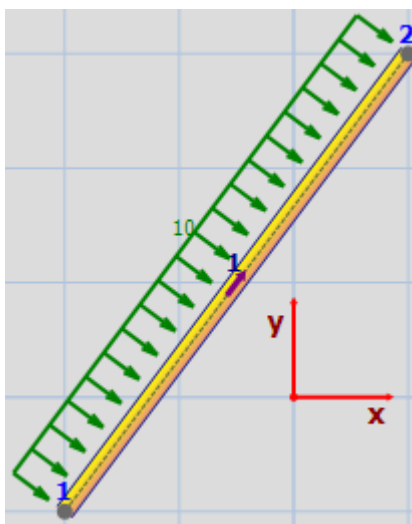
Load value kN/m

OK Cancel

3. Click on the element you want to assign the load to.




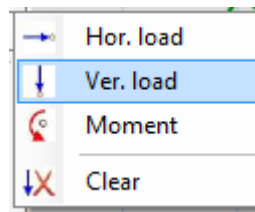
4. The element load has been assigned to the selected element.



Please note that this could also be done from the Frame elements tab on the left program menu, under the loads page.

Applying loads on a node

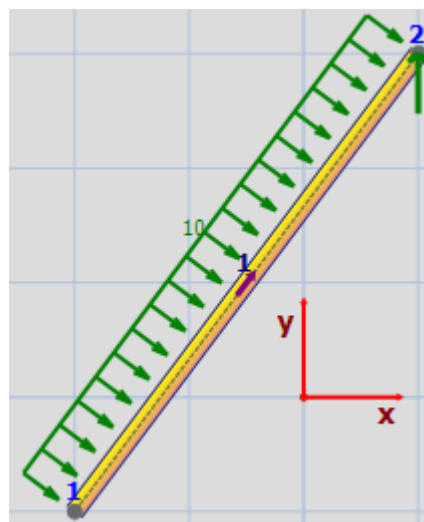
1. Click on the  icon and select from the following menu.



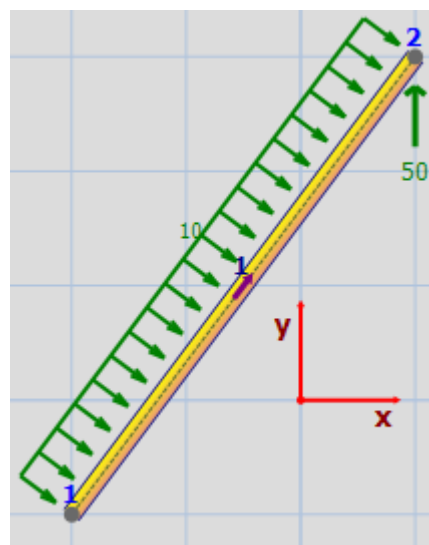
2. Enter a load value in the following form that will come up.

Load value kN

3. Click on the node you want to assign the load to.




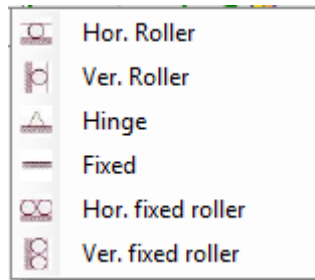
4. The element load has been assigned to the selected node.



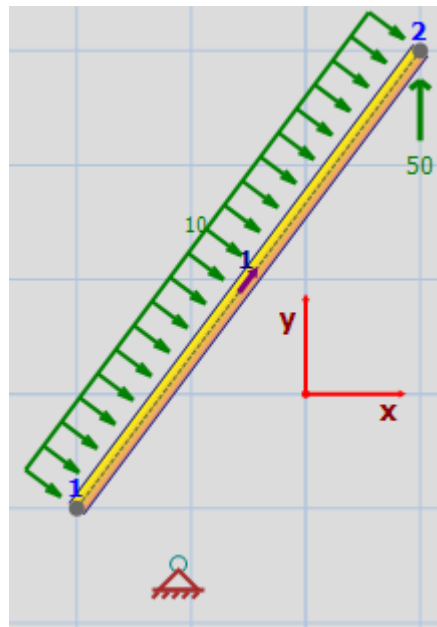
Please note that this could also be done from the Auto generated nodes tab on the left program menu.

Applying restraints (supports) on a node

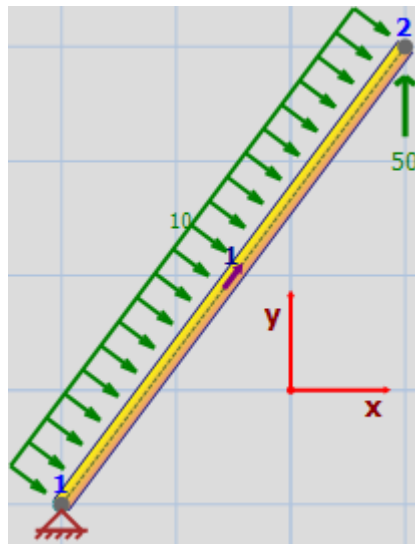
1. Click on the  icon and select the desired support type from the following menu.



2. Click on the node which you want to apply the restraint to.




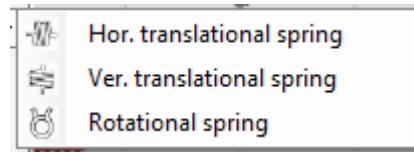
3. Restraint has been assigned to the selected node.



Please note that this could also be done from the Auto generated nodes tab on the left program menu.

Applying spring restraints (supports) on a node

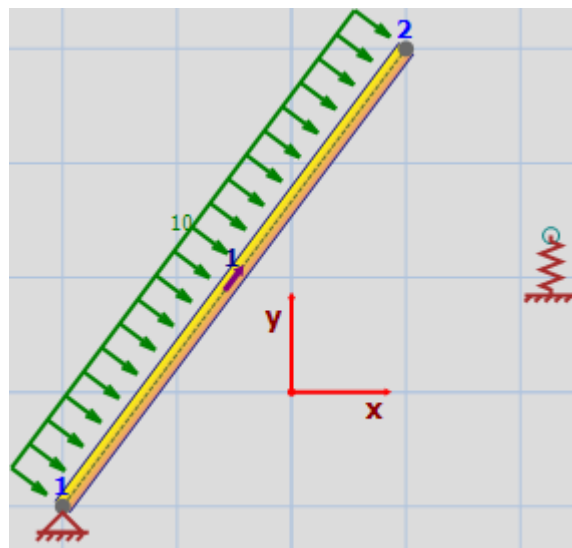
1. Click on the  icon and select from the following menu.



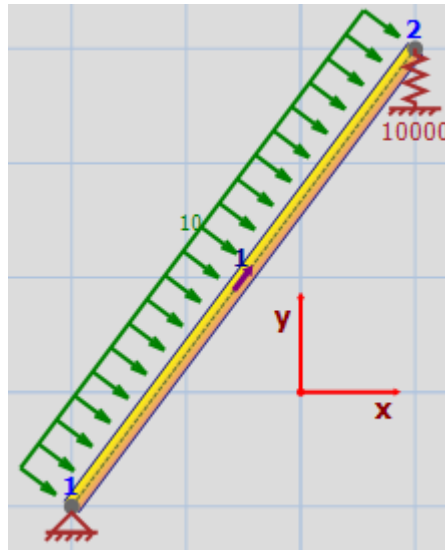
2. Enter the spring stiffness in the following form that will come up.



3. Click on the node you want to apply the spring support to.



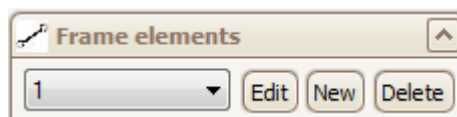
4. Spring support has been assigned.



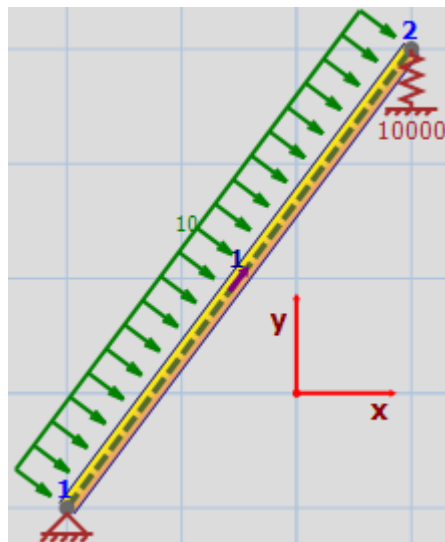
Please note that this could also be done from the Auto generated nodes tab on the left program menu.

Deleting and modifying a frame element

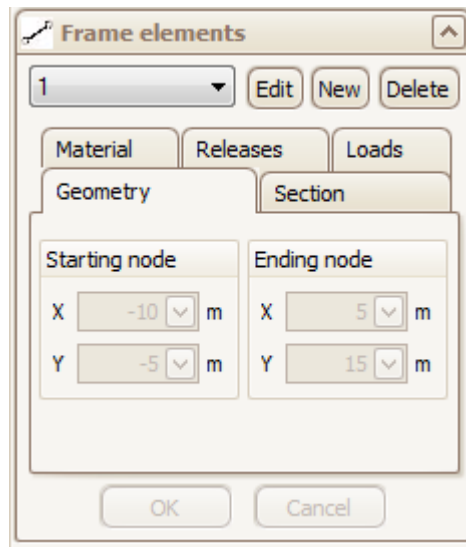
1. Either select the element graphically and double click on it (or right click), or select it from the frame elements combo box in the frame elements menu on your left.



OR (double click or right click)



2. The menu with the examined frame element's data will show, as following:

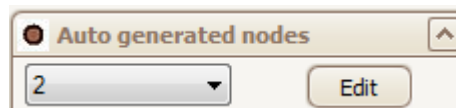


The 'Frame elements' dialog box is shown. It has a title bar with a small icon and the text 'Frame elements'. Below the title bar is a dropdown menu showing '1' and three buttons: 'Edit', 'New', and 'Delete'. Below these are three tabs: 'Material', 'Releases', and 'Loads'. Below the tabs are two sections: 'Geometry' and 'Section'. The 'Geometry' section is active and contains two sub-sections: 'Starting node' and 'Ending node'. The 'Starting node' section has two rows: 'X' with a value of '-10' and 'Y' with a value of '-5', both followed by a unit 'm'. The 'Ending node' section has two rows: 'X' with a value of '5' and 'Y' with a value of '15', both followed by a unit 'm'. At the bottom of the dialog are two buttons: 'OK' and 'Cancel'.

3. Click on Edit in order to enable the controls and start modifying the data, or on Delete to delete the selected frame element.
4. Then click on OK to finish.

Modifying a node

1. Either select the node graphically and double click on it (or right click), or select it from the nodes combo box in Auto generated nodes menu on your left.



The 'Auto generated nodes' dialog box is shown. It has a title bar with a small icon and the text 'Auto generated nodes'. Below the title bar is a dropdown menu showing '2' and a button labeled 'Edit'.

2. The menu with the examined node's data will show, as following:

Auto generated nodes ⬆

2 Edit

Coordinates

X m Y m

Nodal Loads

Px kN

Py kN

Mz kN m

Supports

Degree of freedom	Full support	Spring constant
<input type="checkbox"/> X Translation	<input type="checkbox"/>	<input type="text" value="0"/> kN / m
<input type="checkbox"/> Y Translation	<input type="checkbox"/>	<input type="text" value="10000"/> kN / m
<input type="checkbox"/> Z Rotation	<input type="checkbox"/>	<input type="text" value="0"/> kN m / rad

Initial displacements

Degree of freedom	Initial displacements/rotation
<input type="checkbox"/> X Translation	<input type="text" value="0"/> m
<input type="checkbox"/> Y Translation	<input type="text" value="0"/> m
<input type="checkbox"/> Z Rotation	<input type="text" value="0"/> rad

Initial velocities for dynamic analysis

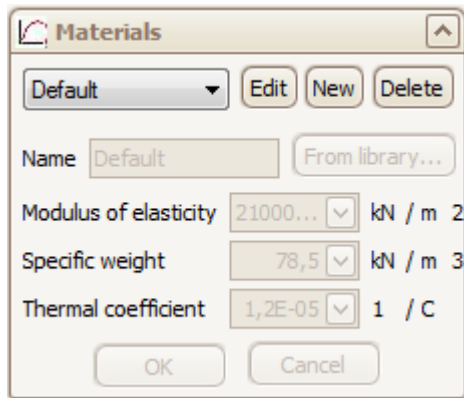
Degree of freedom	Initial velocities
<input type="checkbox"/> X Translation	<input type="text" value="0"/> m /sec
<input type="checkbox"/> Y Translation	<input type="text" value="0"/> m /sec
<input type="checkbox"/> Z Rotation	<input type="text" value="0"/> rad /sec

OK Cancel

3. Click on Edit in order to enable the controls and start modifying the data.
4. Then click on OK to finish.

Creating, modifying and deleting a material

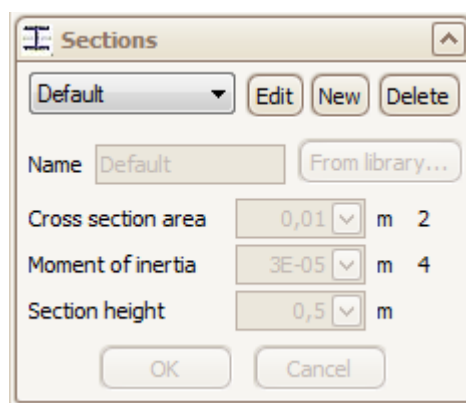
1. Select the material from the materials combo box.



2. Click on Edit to modify its data, on Delete to delete it, or on New button to create a new material.
3. Then click on OK to finish.

Creating, modifying and deleting a section

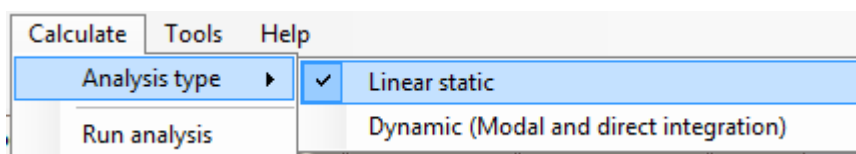
1. Select the section from the materials combo box.




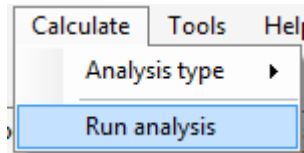
2. Click on Edit to modify its data, on Delete to delete it, or on New button to create a new section.
3. Then click on OK to finish.

Running static analysis

1. Make sure that the option Linear static analysis is checked under the Calculate menu.

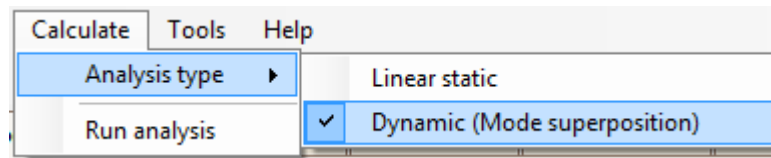


2. Simply click on Run analysis item or on the  toolbar icon.

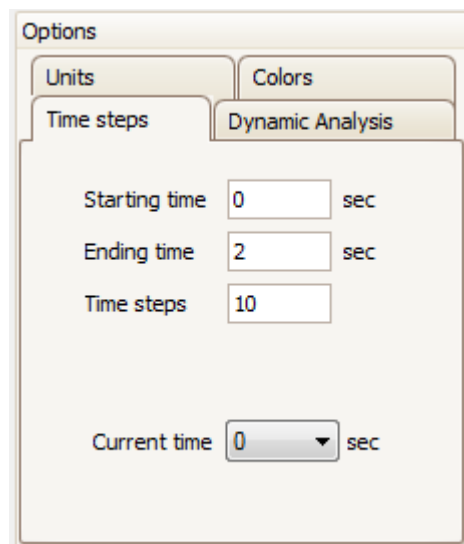


Running dynamic analysis

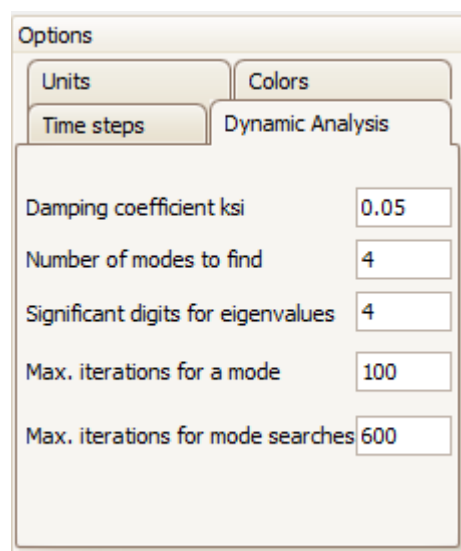
1. Make sure that the option Dynamic (Mode superposition) analysis is checked under the Calculate menu.




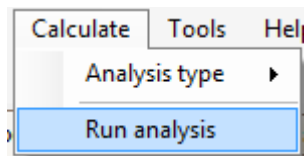
2. Enter time step number in the following tab on your right.



3. Make sure that the preferences under the Dynamic Analysis tab are correct.



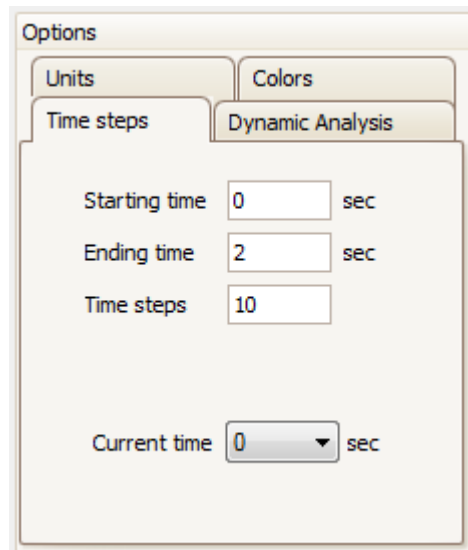
4. Simply click on Run analysis item or on the  toolbar icon.



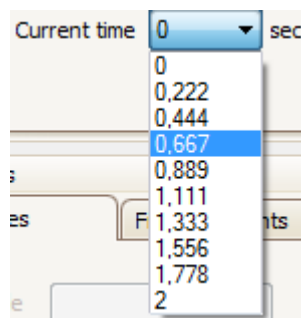
Applying dynamic loads

This can be done very simply following these steps:

1. Set the time step number under the Time step tab on your right.



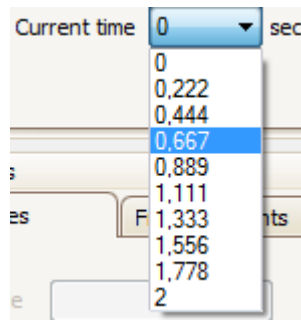
2. Choose the time at which you want to impose a load in your model.



3. Apply the loads in your model as you would do in case of a static analysis. These loads will be imposed only at the selected time (at 0.667 seconds)
4. Continue applying nodal and element forces for each desired time step.

Viewing results of dynamic analysis


1. Choose the time at which you want to see the results.

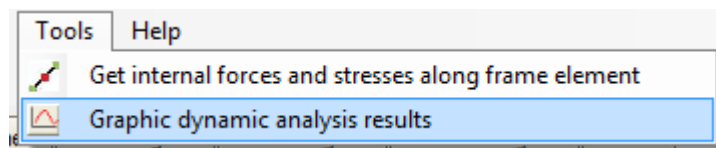


2. All model results will now refer to the selected time step.

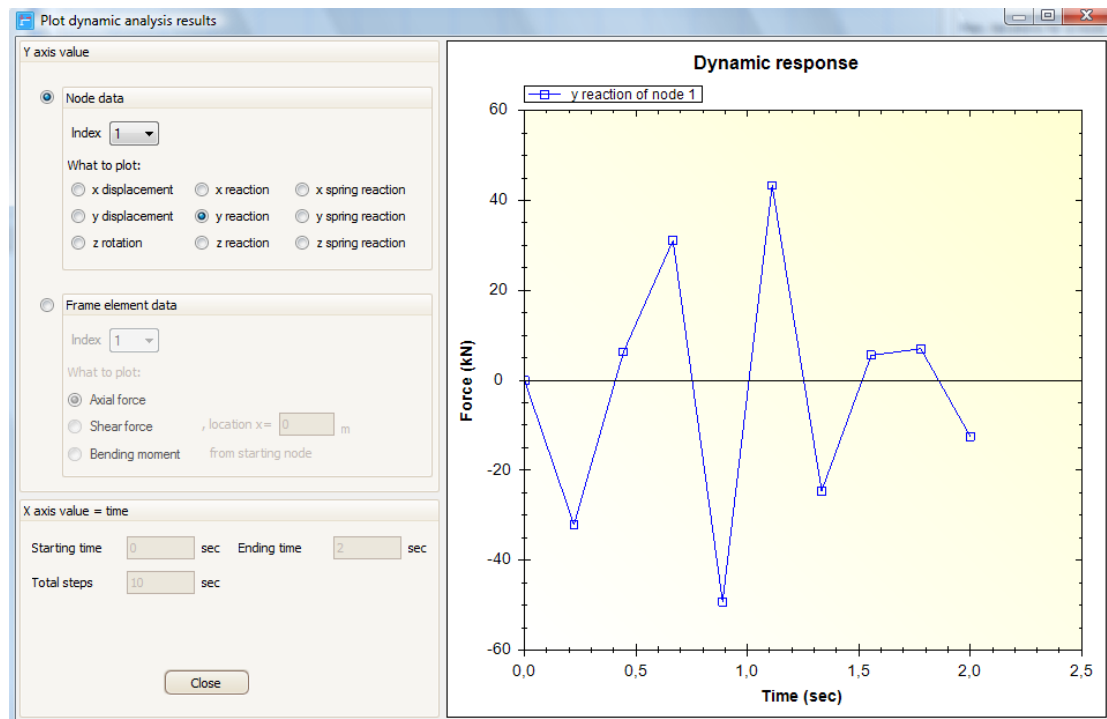
Viewing graphs of dynamic analysis results

You can watch the dynamic response of the model in the whole given time span.

1. After dynamic analysis is complete, click on the  icon or on the Graphics dynamic analysis results item under the Tools menu.




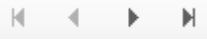


2. The following form is displayed.



3. You can choose among many characteristic values (nodal displacements, reactions, element internal forces etc) which will represent the Y axis. The X axis of the graph is the time.

4. The graph, which is automatically created, shows the behavior of the model in the time span in a compact form. This is of course very useful.

Creating an analysis report

1. Click on the  icon.
2. A window with the created report will display.
3. By using the buttons , you can browse your report. As you can see it contains all model data and results, as well as all diagrams regarding the deformed shape, the internal forces diagrams (axial – shear force and bending moment) and the mode shapes.
4. You can export the report to many formats, including .rtf (Word) and .xls (Excel) file format by clicking on the  icon.
5. The report can be printed by clicking on the  icon.

In case of dynamic analysis, the results in the report refer to the latest time step.